Jordan Journal of Civil Engineering, Volume **1**, No. **2**, **2007**

Development of Finite Element Code for Analysis of Reinforced Concrete Slabs

***M. M. Smadi*$^{1)}$ *and K. A. Belakhdar*$^{2)}$**

$^{1)}$ Associate Professor, Department of Civil Engineering, Faculty of Engineering, Jordan University of Science and Technology, P.O. Box (3030), Irbid 22110, Jordan.

$^{2)}$ Former Graduate Student, Department of Civil Engineering, Faculty of Engineering, Jordan University of Science and Technology, P.O. Box (3030), Irbid 22110, Jordan. Corresponding Author, Fax (00962-2) 7095018, e-mail: msmadi@just.edu.jo

ABSTRACT

A nonlinear finite element code has been developed to suite the analysis of normal and high strength concrete slabs. The computer program was built up using two computer languages, where the program interface part was coded in Visual-Basic language, while the main part was coded using FORTRAN language. A software called NLFEAS (Non-Linear Finite Element Analysis of Slabs) was developed to predicate and study the three dimensional response of reinforced concrete slabs of different grades, variables and boundary conditions under monotonically increasing loads. Using symmetry, a segment representing one quarter of the slab was considered in the analysis which was modeled using 20-node isoparametric brick elements and a 27-integration rule (3*3*3). Proper numerical material models for cracked concrete were incorporated in the analysis. The efficiency and accuracy of the developed code was verified through comparison with available test data, which showed good agreement. The effects of some finite element parameters such as mesh refinement and integral rule were also investigated.

Keywords: Nonlinearity, Finite Element Analysis, Material Modeling, Concrete Slabs, NLFEAS Code, Preprocessor Layer, Postprocessor Layer.

INTRODUCTION

Within the framework of developing advanced design and analysis methods for modern structures the need for experimental research continues. Experiments provide a firm basis for design equations, which are invaluable in the preliminary design stages. Experimental research also supplies the basic information for finite element models, such as material properties. In addition, the results of finite element models have to be evaluated by comparing them with experiments of full-scale models of structural sub-assemblages or, even, entire structures. The development of reliable analytical models can, however, reduce the number of required test specimens for the solution of a given problem, recognizing that tests are time-consuming and very expensive and often do not simulate exactly the loading and support conditions of the actual structure (Marzouk and Jiang, 1996).

Because of the complex behavior of concrete structures, engineers have relied in the past on empirical formulas for the design of concrete elements, which were derived from numerous experiments. With the advent of digital computers and powerful methods of analysis, such as the finite element method, many efforts to develop analytical solutions which would obviate the need for experiments have been undertaken by investigators (Jeyamohan, 1987; Sankarasubramanian and

---

Received on 1/1/2007 and Accepted for Publication on 1/4/2007.

Rajasekaran, 1996; Wang and Hsu, 2001; Hallgren and Bjerke, 2002). The finite element method has thus become a powerful computational tool, which allows complex analyses of the nonlinear response of RC structures to be carried out in a routine fashion. With this method, the importance and interaction of different nonlinear effects on the response of RC structures in general and slabs in particular can be analytically studied (Chen and Saleeb, 1982; Parton and El-Barbary, 1983; Vidosa et al., 1988; Marzouk and Chen, 1993; Marzouk and Jiang, 1996; Al-Nasra, 1997; Jiang and Mirza, 1997; Reitman and Yankelevsky, 1997; Polak, 1998; Huang et al., 1999; Enochsson and Dufvenberg, 2001; Shanmugam et al., 2002; Vainiunas et al., 2004; Polak, 2005; Murray et al., 2005; Deaton, 2005).

Table 1: Properties of investigated slabs.

Slab	Slab thickness *h* [mm]	Effective depth *d* [mm]	Concrete strength *f'c* [N/mm^2]	Steel yielding stress *fy* [N/mm^2]	Steel ratio *ρ* [%]
NS1	120	95	42	496	1.47
HS1	120	95	67	496	0.49
HS2	120	95	70	496	0.84
HS7	120	95	74	496	1.19
HS3	120	95	69	496	1.47
HS4	120	90	66	496	2.37
NS2	150	120	30	496	0.94
HS5	150	95	68	496	0.64
HS6	150	120	70	496	0.94
HS8	150	120	69	420	1.11
HS9	150	120	74	420	1.61
HS10	150	120	80	420	2.33
HS11	90	70	70	496	0.95
HS12	90	70	75	496	1.52
HS13	90	70	68	496	2.00
HS14	120	95	72	496	1.47
HS15	120	95	71	496	1.47

In the present study, a nonlinear finite element code NLFEAS has been developed which can be used for reinforced concrete structures in general and particularly for the analysis of normal and high strength concrete slabs. The developed computer program incorporated two computer languages, namely; FORTRAN and Visual-Basic languages. The main part of the program was coded using FORTRAN language while the interface part was written in Visual-Basic language. The source code was developed from a program called P3DNFEA (Al-Shaarbaf, 1990), which was based on a general nonlinear finite element program NAGFE Library, level 0 (NAGFE Liberary, 1980). The software NLFEAS (Non-Linear Finite Element Analysis of Slabs) was developed to predict the three dimensional behavior of reinforced concrete slabs of different grades and boundary conditions under gravity loads.

The concrete was represented by using a 20-noded isoparametric brick element with a total of sixty degrees of freedom as three translations at each node (u, v, w) in X, Y and Z directions, respectively, and a 27-integration rule (3*3*3). The reinforcing bars were modeled as one dimensional element subjected to axial force only embedded within the concrete brick elements. A perfect bond is assumed to occur between the two materials.

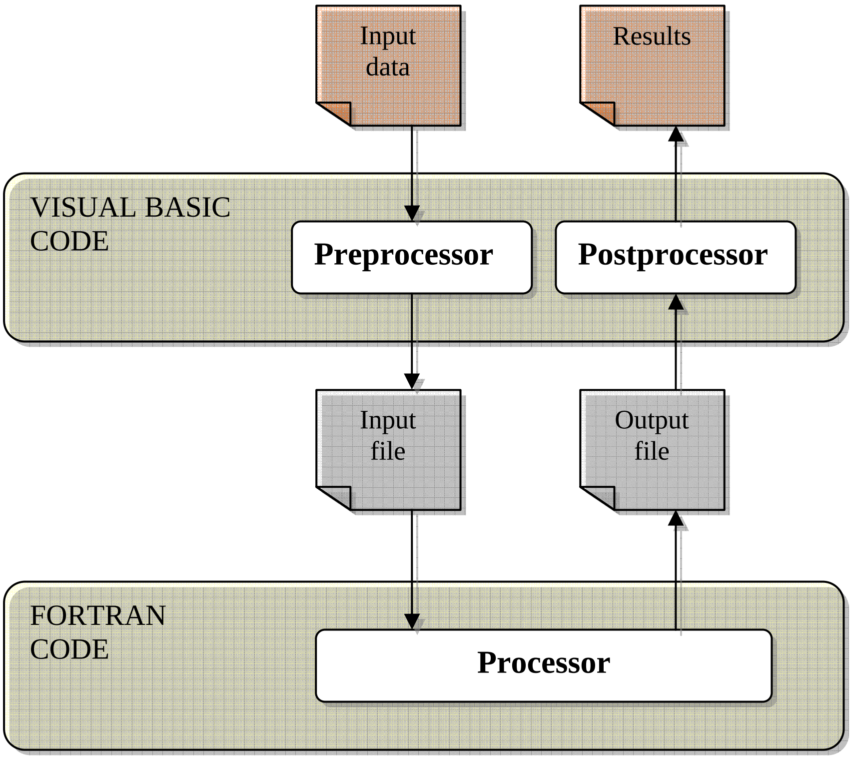


Fig. 1: NLFEAS program organization.

Table 4: Mesh density in terms of time consumed.

Mesh	**Time consumed**	**Ratio to the 108-mesh**
3x3x2 (18-elements)	$00^{mn}:40^{s}$	3%
4x4x3 (48-elements)	$04^{mn}:51^{s}$	34%
6x6x3 (108-elements)	$13^{mn}:41^{s}$	100%

The nonlinear behavior of concrete in compression was simulated by an elastic-plastic work-hardening model up to the onset of crushing. A linear-parabolic stress-strain curve has been used to model the equivalent uniaxial stress-strain diagram of both normal and high strength concrete.

In tension, a fixed smeared crack model has been used with a tension-stiffening concept to represent the retained post-cracking tensile stresses. A shear retention model that modifies the shear stiffness, and softening models that reduce the concrete compressive strength, due to cracking are also implemented. The numerical material models will allow for both high and normal strength concrete.

The non-linear equations of equilibrium have been solved using the incremental-iterative technique based on the modified Newton-Raphson method and the equations are updated each tenth iteration. The convergence of the solution was controlled by a force convergence criterion.

One of the main objectives of the study is to verify the efficiency and accuracy of the modified computer program NLFEAS throughout the comparison of the predicted with available experimental measurements. Moreover, parametric studies are carried out to investigate the effect of some important finite element parameters, on the analysis and behavior of reinforced concrete slabs.

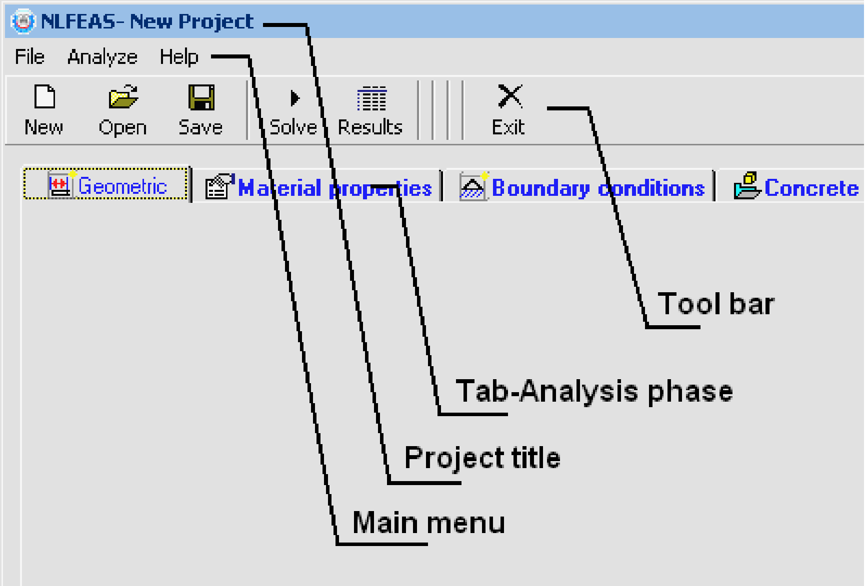
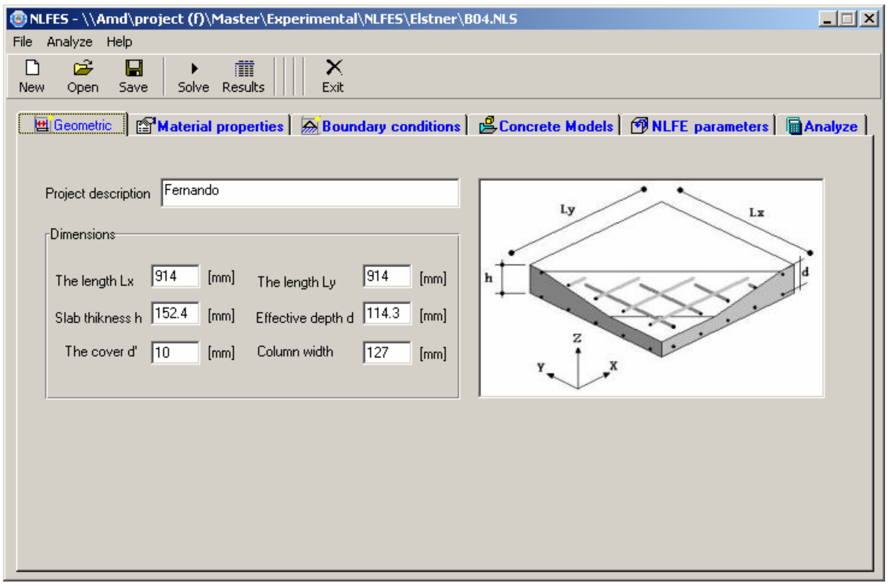


Fig. 2: NLEFAS program interface.



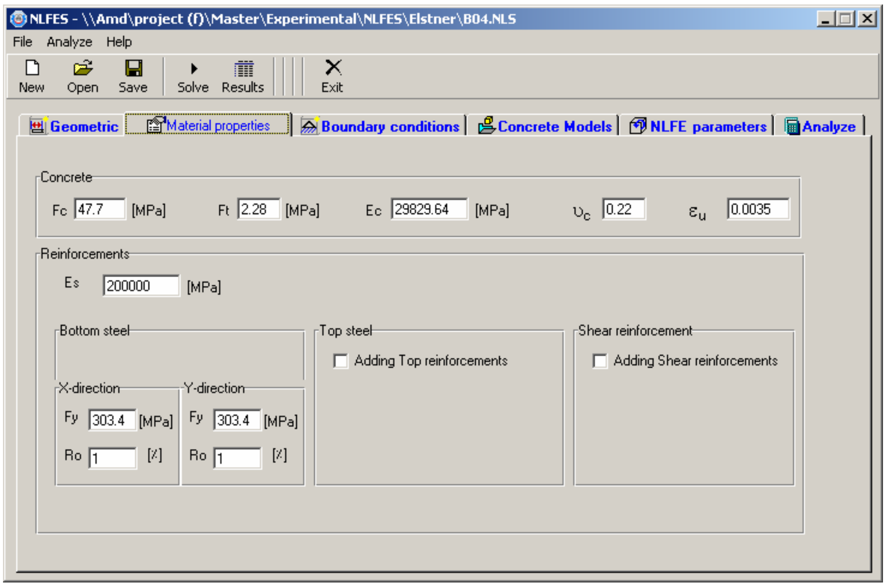
NLFES - \\Amd\project (f)\Master\Experimental\NLFES\Elstner\B04.NLS

Project description: Fernando

Dimensions

The length Lx	914	[mm]	The length Ly	914	[mm]
Slab thikness h	152.4	[mm]	Effective depth d	114.3	[mm]
The cover d'	10	[mm]	Column width	127	[mm]

Fig. 3: Input of geometrical data.



Concrete

Fc 47.7 [MPa]
Ft 2.28 [MPa]
Ec 29829.64 [MPa]
υ_c 0.22
ε_u 0.0035

Reinforcements

Es 200000 [MPa]

Bottom steel

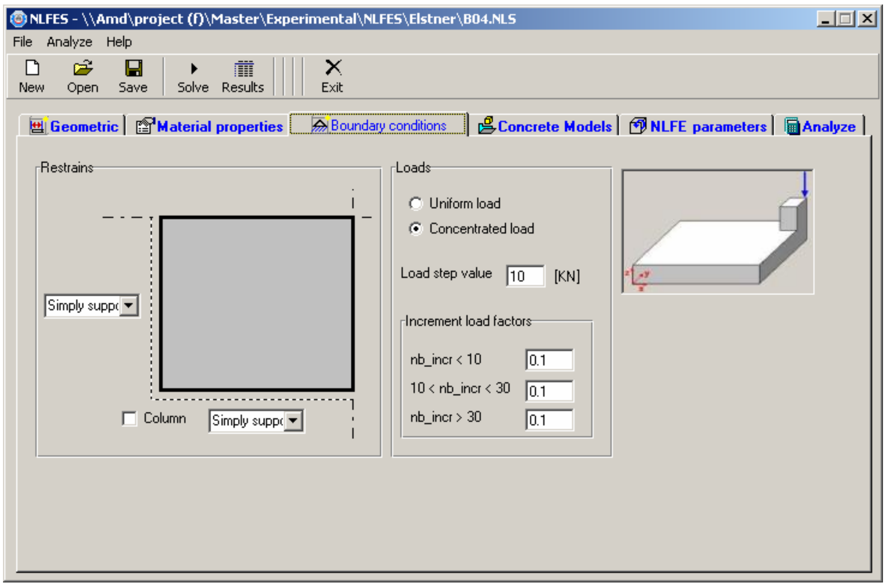
X-direction
Fy 303.4 [MPa]
Ro 1 [%]

Y-direction
Fy 303.4 [MPa]
Ro 1 [%]

Top steel
☐ Adding Top reinforcements

Shear reinforcement
☐ Adding Shear reinforcements

Fig. 4: Input of material properties.



Restrains

Simply suppo
☐ Column
Simply suppo

Loads

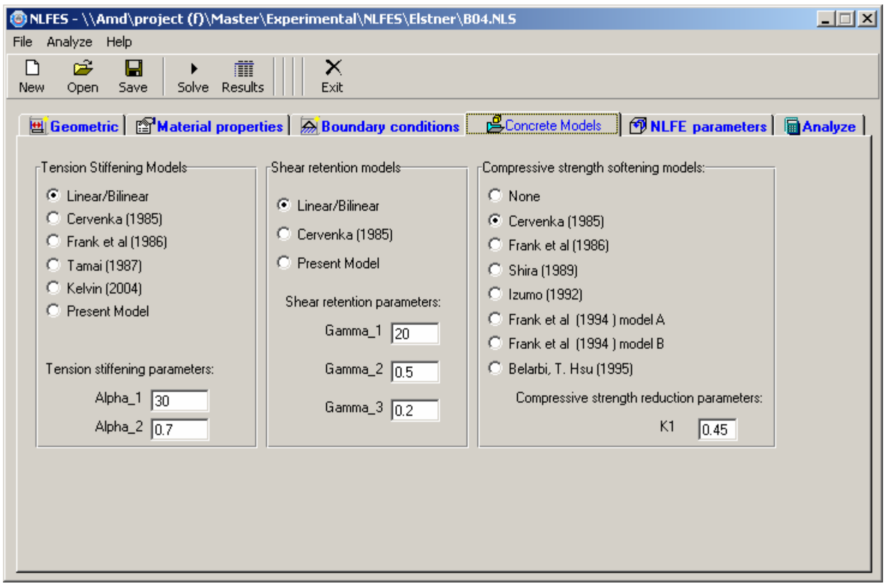
○ Uniform load
● Concentrated load

Load step value 10 [KN]

Increment load factors

nb_incr < 10 0.1
10 < nb_incr < 30 0.1
nb_incr > 30 0.1

Fig. 5: Input of boundary conditions and loads.



Tension Stiffening Models

● Linear/Bilinear
○ Cervenka (1985)
○ Frank et al (1986)
○ Tamai (1987)
○ Kelvin (2004)
○ Present Model

Tension stiffening parameters:

Alpha_1 30
Alpha_2 0.7

Shear retention models

● Linear/Bilinear
○ Cervenka (1985)
○ Present Model

Shear retention parameters:

Gamma_1 20
Gamma_2 0.5
Gamma_3 0.2

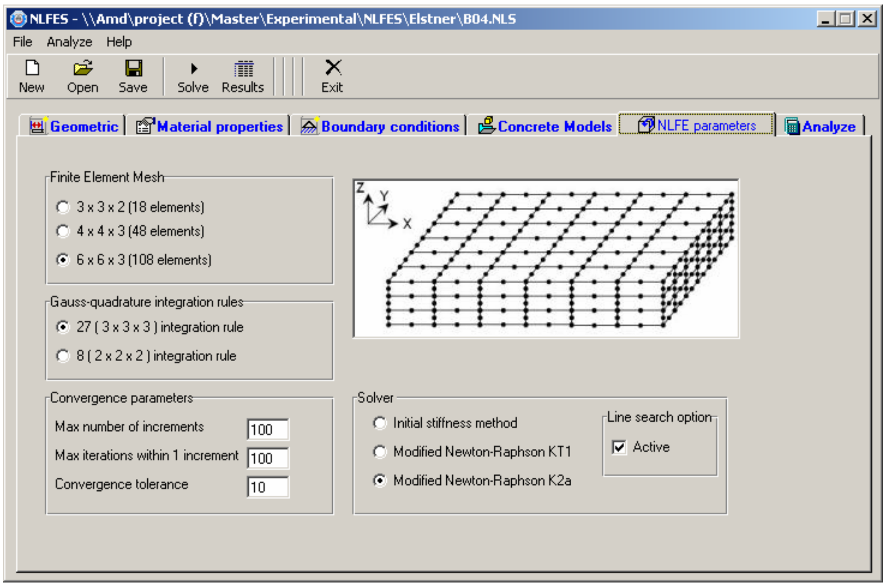
Compressive strength softening models:

○ None
● Cervenka (1985)
○ Frank et al (1986)
○ Shira (1989)
○ Izumo (1992)
○ Frank et al (1994) model A
○ Frank et al (1994) model B
○ Belarbi, T. Hsu (1995)

Compressive strength reduction parameters:

K1 0.45

Fig. 6: Input of concrete numerical models.



Finite Element Mesh

○ 3 x 3 x 2 (18 elements)
○ 4 x 4 x 3 (48 elements)
● 6 x 6 x 3 (108 elements)

Gauss-quadrature integration rules

● 27 (3 x 3 x 3) integration rule
○ 8 (2 x 2 x 2) integration rule

Convergence parameters

Max number of increments 100
Max iterations within 1 increment 100
Convergence tolerance 10

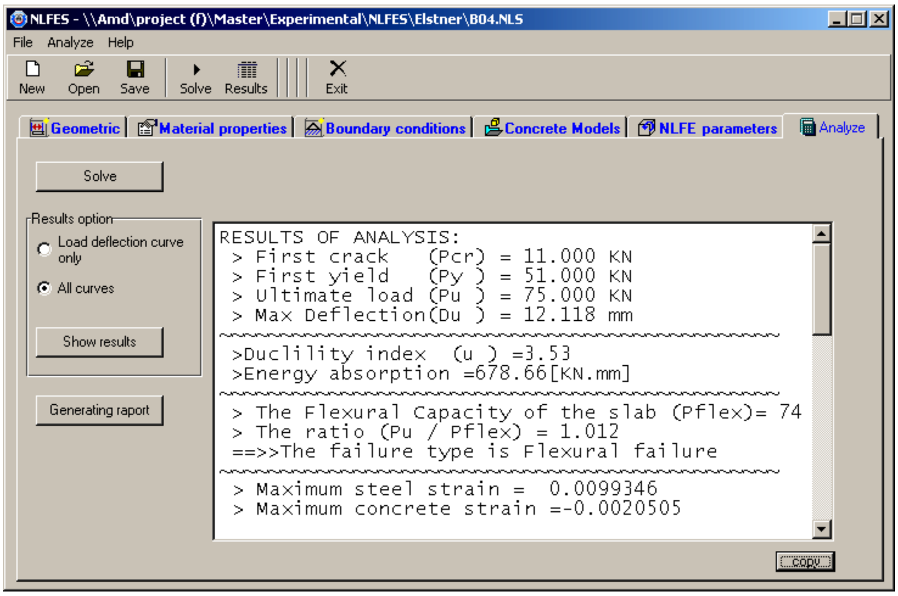
Solver

○ Initial stiffness method
○ Modified Newton-Raphson KT1
● Modified Newton-Raphson K2a

Line search option

☑ Active

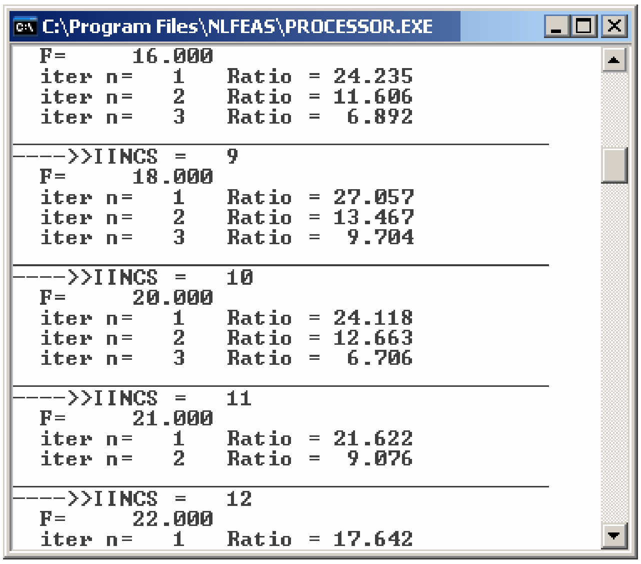
Fig. 7: Input of finite element parameters.



Results option: ○ Load deflection curve only ● All curves

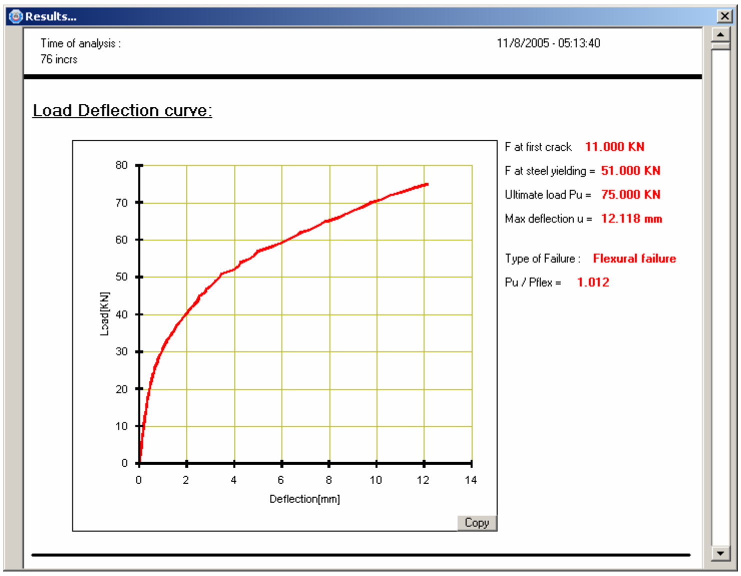
```
RESULTS OF ANALYSIS:
 > First crack   (Pcr) = 11.000 KN
 > First yield   (Py ) = 51.000 KN
 > Ultimate load (Pu ) = 75.000 KN
 > Max Deflection(Du ) = 12.118 mm
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
 >Duclility index  (u ) =3.53
 >Energy absorption =678.66[KN.mm]
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
 > The Flexural Capacity of the slab (Pflex)= 74
 > The ratio (Pu / Pflex) = 1.012
 ==>>The failure type is Flexural failure
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
 > Maximum steel strain =  0.0099346
 > Maximum concrete strain =-0.0020505
```

Fig. 8: Analysis of the problem.



```
    F=      16.000
    iter n=    1    Ratio  = 24.235
    iter n=    2    Ratio  = 11.606
    iter n=    3    Ratio  =  6.892
________________________________________
----->>IINCS  =    9
    F=      18.000
    iter n=    1    Ratio  = 27.057
    iter n=    2    Ratio  = 13.467
    iter n=    3    Ratio  =  9.704
________________________________________
----->>IINCS  =    10
    F=      20.000
    iter n=    1    Ratio  = 24.118
    iter n=    2    Ratio  = 12.663
    iter n=    3    Ratio  =  6.706
________________________________________
----->>IINCS  =    11
    F=      21.000
    iter n=    1    Ratio  = 21.622
    iter n=    2    Ratio  =  9.076
________________________________________
----->>IINCS  =    12
    F=      22.000
    iter n=    1    Ratio  = 17.642
```

Fig. 9: Solving the problem.



```
Time of analysis :                                    11/8/2005 - 05:13:40
76 incrs

Load Deflection curve:

F at first crack       11.000 KN
F at steel yielding =  51.000 KN
Ultimate load Pu =     75.000 KN
Max deflection u =     12.118 mm

Type of Failure :   Flexural failure
Pu / Pflex =     1.012
```

Fig. 10: Partial results (Load-deflection curve).

DESCRIPTION OF THE NLFEAS PROGRAM

The present computer program NLFEAS is a combination of two layers, which are:

1. The main layer or the processor layer
2. The secondary layer which consists of:
   a. a preprocessor
   b. a postprocessor

The flow chart shown in Fig. (1) represents schematically the internal-communications between the program components.

The Main Layer

The main layer represents the principal part of the whole program. In this part, all the general finite element formulation and the constitutive relationships are incorporated, which are coded in FORTRAN computer language. The original program was modified in order to analyze reinforced concrete slabs and to improve the program's accuracy and capability to solve structures with a large number of nodes and elements. Various numerical models including tension stiffening, shear retention and compressive strength softening models were incorporated into the processor layer in order to improve the analytical prediction and enhance its accuracy and efficiency (Belakhdar, 2006).

The Secondary Layer

This layer is coded in Visual-basic language; it consists of two parts; the preprocessor and the postprocessor.

The Preprocessor Part

The processor layer required a very complex data file format with a huge number of data especially when fine mesh is used. Besides, this format was very difficult to formulate which makes the operation of checking errors more complicated. Moreover, the input file format may require a global reconstruction when structure geometric properties change.

Thus, the main objective of the preprocessor is to facilitate the use of the program and make it easy to change any structural properties without difficulties, and reduce the required data to the minimum limits. These operations were done by developing computer sub-routines capable of collecting basic input data, and treat them in order to generate the input file to contain full information required for the processor layer.

The Postprocessor Part

After the structure is solved by the processor layer, a large number of output files are obtained containing a very large amount of results such as, nodal displacements, concrete normal and shear stresses and strains, steel stress and strains. Hence, the postprocessor treats all output data by self-searching, sorting and selecting only the required data such as the maximum deformations, maximum stresses and strains at specific nodes.

After these operations of treatments of the output results, the final results are listed and plotted graphically as follows:

- Load-deflection curve at center of the slab up to failure load.
- Top concrete strain distribution curves at cracking, yielding and ultimate load.
- Bottom steel stress and strain distribution curves at cracking, yielding and ultimate load.
- Slab curvature variations along the slab span at cracking, yielding and ultimate load.
- Additional results may also be obtained such as ductility of the analyzed slab, energy absorption and other data.

Analysis Termination Criteria

Generally, the collapse of any structure under load control can be indicated when no further loading can be carried out. In the nonlinear finite element analysis, it was observed that the number of iteration increases rapidly to achieve the convergence when the applied load is in close proximity to the failure load. In that range the iterations become unstable and may never be achieved. Consequently, it is necessary to specify a suitable analysis termination criterion to stop the analysis in such cases. Usually the continuation of analysis is limited by specifying a maximum number of iteration, maximum number of increments and maximum deflection. But these criteria are not true all the time and false predictions may be obtained. In such cases, the solution can not converge even under small loads due to the large variation of the structure stiffness caused by sudden cracks formation or steel yielding especially in complex thin structures (Al-Shaarbaf, 1990). Moreover, the solution may not converge if a very small tolerance was selected if very large number of increments was specified.

In finite element analysis, several criteria are used to control the analysis, such as controlling the maximum specified strain or stress, or limiting the maximum displacement which can occur in the current structure, or specifying an upper limit for a number of increments and iterations.

In the present study the nonlinear finite element analysis stops when any of the following criteria is satisfied:

- The stiffness matrix is no longer definitely positive.
- The determinant of the stiffness matrix is null.
- The number of increments exceeds the maximum specified number.
- The number of iterations exceeds the maximum specified number.
- The Collapse of a structure takes place (the maximum equivalent total strain is reached), where in this case, the last applied load is considered as the ultimate load capacity of slab.

Step-by-step Analysis of Reinforced Concrete Slab

1- New/Open/Save projects is accessible in both tool bar or [file] in the main menu as shown in Fig. (2).

2- Definition of the geometric properties: the geometric slab properties may be defined in [Geometric]-Tab, noting that only one quarter is taken into consideration as shown in Fig. (3).

3- Definition of the material properties: concrete and steel material properties may be defined in [material properties]-Tab as shown in Fig. (4).

4- Applying the slab boundary conditions and loads: in [Boundary conditions] both applied loads and support conditions may be defined. Any support may be defined as: continuous, fixed, simply supported or free edge. Uniform or concentrated loads are the accessible choices, as shown in Fig. (5).

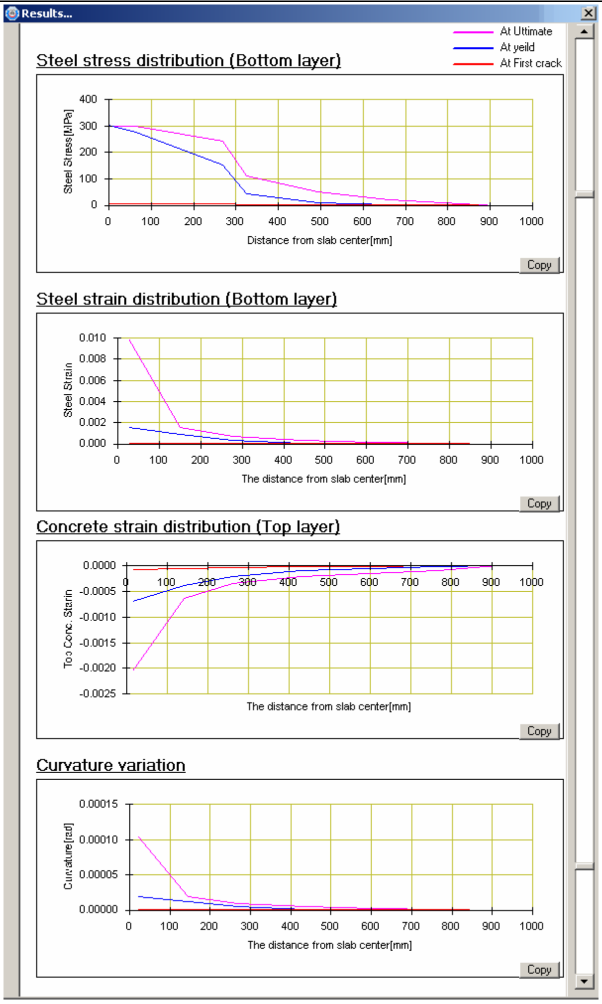


Fig. 11: Full results (Stress, strain, curvature distribution).

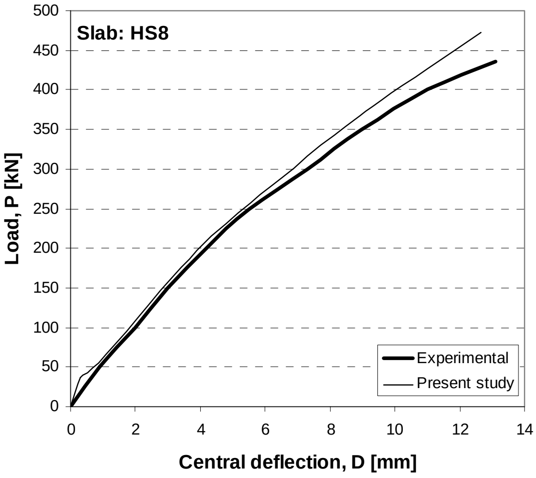


Fig. 12: Comparison of load-deflection of slab HS8.

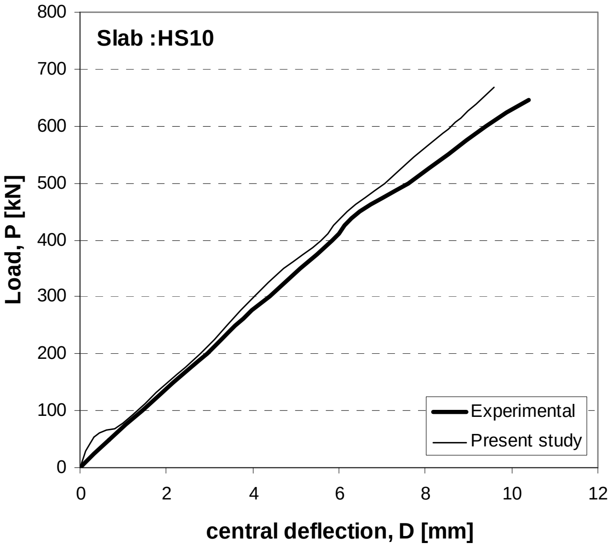


Fig. 13: Comparison of load-deflection of slab HS10.

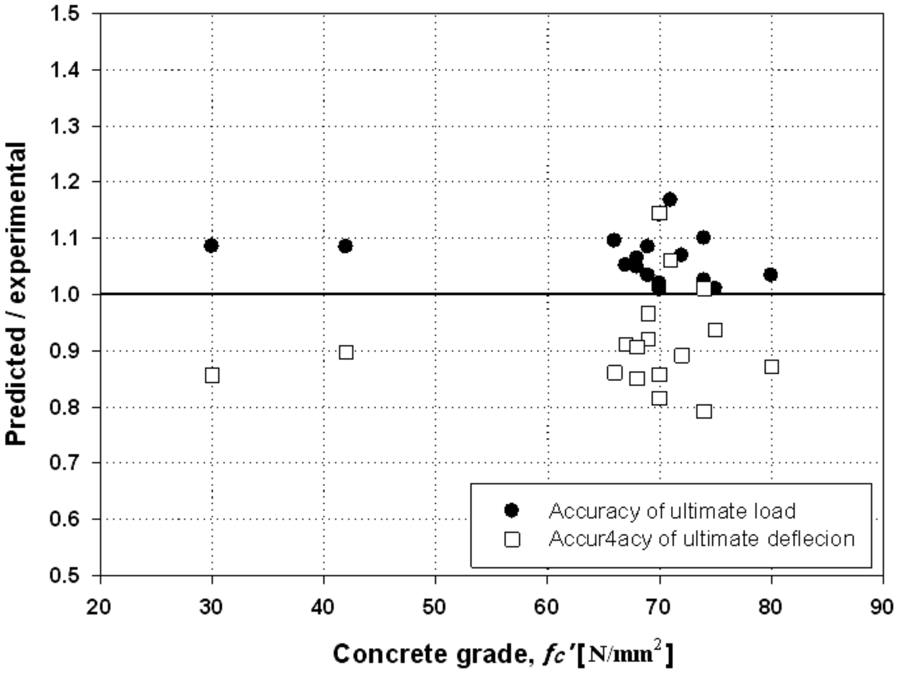


Fig. 14: Accuracy of the predicted results in terms of concrete grade.

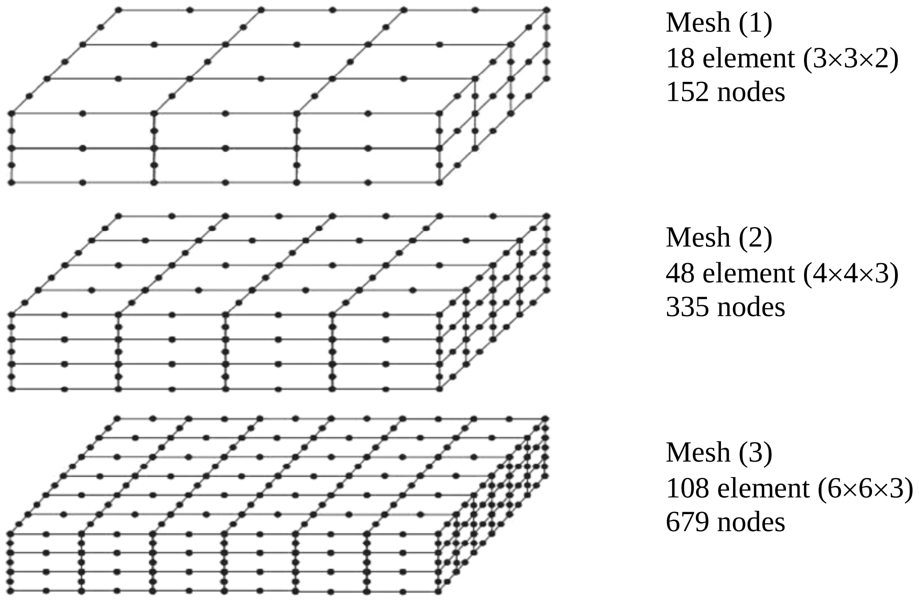


Fig. 15: Finite element meshes.

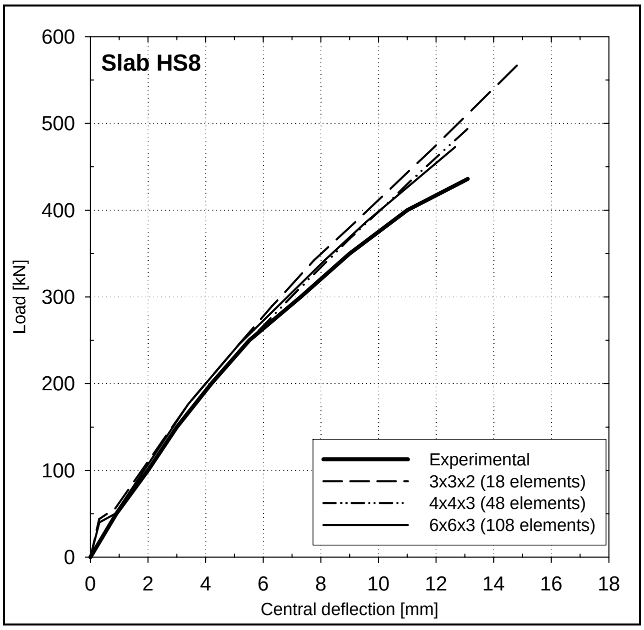


Fig. 16: Effect of mesh refinement on the numerical analysis accuracy.

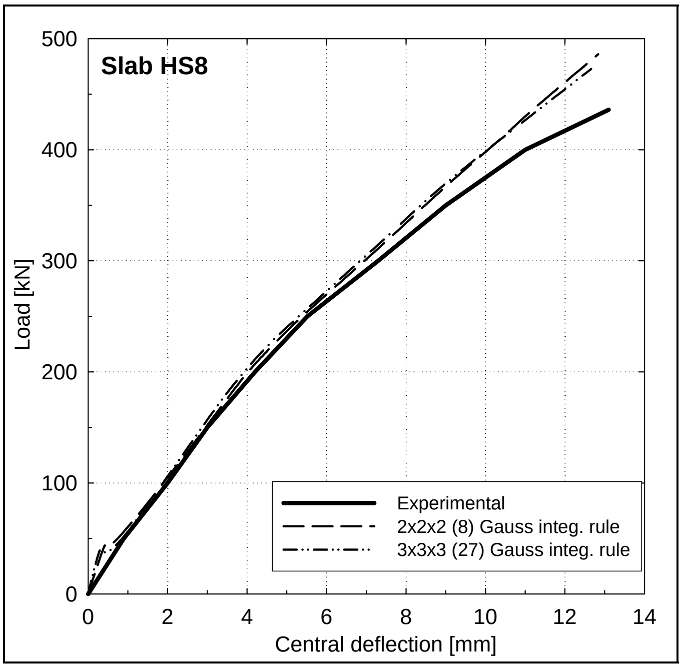


Fig. 17: Effect of integration rule on the numerical analysis.

5- Selection of the concrete numerical models: many numerical models are available for selection as shown in Fig. (6). Some models may be used to calibrate the predicted results according to experimental results such as: the bilinear model in both tension stiffening and shear retention concept in addition to the compressive strength softening models such as those given by (Cervenka, 1985; Vecchio and Collins, 1986; Vecchio et al., 1994; Belarbi and Hsu, 1995 and others).

6- Definition of finite element parameters: the mesh density, integration rule and other finite element parameters may be selected and defined in [NLFE parameters]-Tab as shown in Fig. (7). Note that, the finer mesh consumes longer time than the coarse one. The line search method helps the solution to converge well, so it is recommended that this option is kept active.

7- Analyzing and obtaining the results: [Solve] button in [Analyze]-Tab or in the tool bar may be clicked to start the analysis as shown in Fig. (8). If there are no errors, the program starts analyzing the problem as shown in Fig. (9).

8- Additional graph: After the analysis, partial or full results (curves as shown in Figs. (10) and (11)) may be obtained by selecting the appropriate [Results option] then clicking on [Show results] button or [Results]-Button in the tool bar. Finally, generation of small report contains the most important results which may be obtained by clicking on [Generation report] button in [Analyze]-Tab as shown in Fig. (8).

VALIDATION OF THE ANALYTICAL RESULTS

From the available literature concerning the experimental work on reinforced concrete slabs, seventeen slabs were selected to be used to validate and corroborate the predicted analytical results, with taking into consideration the variety of the different slab parameters. The experimental tests carried out by Marzouk and Hussein (1991) on normal and high strength concrete slabs were used in the analysis. The properties of the investigated slabs are given in Table (1). The predicted results, in terms of deflection and load at yielding of steel reinforcement and at failure obtained using the NLFEAS program, were compared with the experimental measurements, as given in Table (2).

For all analyzed slabs, the predicted finite element load deflection responses were generally very close to the experimental measurements. Typical load deflection curves are shown in Figs. (12) and 13 for slabs HS8 and HS10, respectively. The ratios of predicted to experimental ultimate deflection and load values are shown in Fig. (14). According to Table (2) and Fig. (14), it can be observed that the present FE analysis provides a reasonable agreement with the measured ultimate values of load and deflection. The predicted to experimental ultimate load ranges from 1.008 to 1.167 with average ratio of 1.066 and standard deviation of 0.045, while the predicted ultimate deflection to experimental results ranges from 0.792 to 1.145 with average ratio of 0.921 with standard deviation of 0.090. Thus, the present FE analysis performs satisfactorily.

The steel strains and ductility of slabs and their energy absorption are evaluated and compared with the experimental results, as given in Table (3). Where the ductility is identified as the ratio of the deflection at failure to that at steel yielding, and the energy absorption as the area under the load-deflection curve (Marzouk and Hussein 1991). Again, good agreement between predicted and experimental values is observed for most slabs.

Consequently, the current modified program NLFEAS may be described as stable, accurate and suitable for analytical analysis of reinforced normal and high strength concrete slabs.

PARAMETRIC STUDY OF FINITE ELEMENT PARAMETERS

A parametric study is conducted to investigate the effect of some finite element parameters on the behavior of reinforced concrete slabs. To demonstrate the sensitivity of the total load-deflection curve to the variation of a specific parameter, all other parameters are kept constant at an optimal value.

The Effect of Mesh Refinement

In order to investigate the effect of different finite element meshes on the accuracy of the predicted results, numerical response and time consuming three different meshes were considered, while all other finite element parameters such as the integration rule and tolerance...etc. were maintained unified, as follows (Fig. 15):

- 18 elements (3×3×2).
- 48 elements (4×4×3).
- 108 elements (6×6×3).

The analytical solutions obtained using the different meshes are compared with the experimental test results as shown in Fig. (16). Results of the numerical analysis reveal that the load-deflection curves obtained using 18 elements mesh has a response slightly stiffer with less accurate ultimate load and corresponding deflection than those obtained using other meshes. As generally the case, better agreement with an experimental test is obtained using the finest mesh (108 elements). Noting that the analysis was carried out using a PC computer with AMD processor (1.8 GHz), the time consumed of each mesh was compared together by taking the 108 elements consumed time as reference to eliminate the Processor's speed. The results are listed in Table (4).

Effect of Integral Rule

Two different types of integration rules were used to test its influence on the accuracy of the predicted results and efficiency (time consuming), namely; 8-point and 27-point integration rule. The resulting finite element results obtained for both integration rules are shown in Fig. (17). It can be noticed that the overall load-deflection behavior obtained using both rules was almost identical, but the curve obtained using 8-IR is slightly less stiffer and less accurate than that obtained using 27-IR starting from the post cracking stage. It was also observed that 8-IR was less stable during the analysis especially after the cracking stage and before the failure, than those obtained using 27-IR. This phenomenon may be due to the fact that at onset of cracking, or when steel yields at one integration point, all other integration points in the element are affected at the next iteration (Kwak and

Filippou, 1990). However, the ultimate load and corresponding deflection predicted using 27 and 8-integration rules have small differences.

CONCLUSIONS

Based on the current study, the following conclusions can be made:

1. Proper nonlinear finite element analysis may satisfactorily and accurately predict the overall behavior of normal and high strength concrete slabs including load-deflection response, stresses, strains, ductility and energy absorption.
2. The present FE analysis was found to perform satisfactory for all analyzed slabs, where the predicted response and the experimental data were generally very close.
3. The current modified program NLFEAS may be described as stable, accurate and suitable for analysis of reinforced normal and high strength concrete slabs.
4. Before conducting an analytical investigation, the calibration of the finite element models must be carried out and verified through comparison of the predicted results with trusty experimental data.
5. The mesh density has a significant effect on the predicted results accuracy and is time consuming, hence a moderate mesh density may by the best choice.
6. The use of 8(2x2x2) integration rule in large concrete brick elements may give less accurate results and sometimes be overestimated because it may suffer from the instability (convergence not always achieved) due to cracking of concrete or steel yielding. However, the 27 (3x3x3) integration rule is more suitable, accurate and stable as well.

REFERENCES

Al-Nasra, M. 1997. Finite Element Analysis of Floor Slabs under Warping Effect, *Engineering and Structures*, 19 (7): 533-539.

Al-Shaarbaf, I. 1990. Three-Dimensional Non-Linear Finite Element Analysis of Reinforced Concrete Beams in Torsion, PhD Thesis, University of Bradford.

Belakhdar, K.A.M. 2006. Three Dimensional Nonlinear Finite Element Analysis of High Strength Reinforced Concrete Slabs, MSc Thesis, Faculty of Graduate Studies, Jordan University of Science and Technology, Irbid, Jordan, 188.

Belarbi, A. and Hsu, T.T.C. 1995. Constitutive Laws of Softened Concrete in Biaxial Tension-Compression, *ACI Structural Journal*, 92 (5): 562-573.

Cervenka, V. 1985. Constitutive Model for Cracked Reinforced Concrete, *ACI Journal*, 82 (8): 882-877.

Chen, W.F. and Saleeb, A.F. 1982. Constitive Equations for Engineering Material, John Wiley and Sons, New York, 1, 580.

Deaton, J.B. 2005. A Finite Element Approach to Reinforced Concrete Slab Design, MSc Thesis, School of Civil and Environmental Engineering, Georgia Institute of Technology, 170.

Enochsson, O. and Dufvenberg, P. 2001. Concrete Slabs Designed with Finite Element Methods, MSc Thesis, Department of Civil and Mining Engineering, Lulea University of Technology, Sweden, 146.

Hallgren, M. and Bjerke, M. 2002. Non-linear Finite Element Analyses of Punching Shear Failure of Column Footings, *Cement and Concrete Composites*, 24: 491-496.

Huang, Z., Burgess, L. and Plank, R. 1999. Nonlinear Analysis of Reinforced Concrete Slabs Subjected to Fire, *ACI Structural Journal*, 96 (1), 127-136.

Jeyamohan, C. 1987. Non-linear Finite Element Analysis of Reinforced Concrete Members, MSc Thesis, Faculty of the Graduate School of the New Jersy Institute of Technology, 128.

Jiang, J. and Mirza, F. 1997. Nonlinear Analysis of Reinforced Concrete Slabs by Discrete Finite Element Approach, *Computers and Structures*, 65(4): 1667-1687.

Kwak, H.G. and Filippou, F.C. 1990. Finite Element Analysis of Reinforced Concrete Structures under Monotonic Loads, Structure Engineering Mechanics and

Materials, Report No.USB/SEMM-90/14, Department of Civil Engineering, University of California, Berkely, 120.

Marzouk, H. and Chen, Z. 1993. Finite Element Analysis of High Strength Concrete Slabs, *ACI Structural Journal*, 90 (5): 505-513.

Marzouk, H. and Hussein, A. 1991. Experimental Investigation on the Behavior of High Strength Concrete Slabs, *ACI Structural Journal*, 88 (6): 701-713.

Marzouk, H. and Jiang, D. 1996. Finite Element Evaluation of Shear Enhancement of High Strength Concrete Plates, *ACI Structural Journal*, 93 (6): 667-673.

Murray, K.A., Cleland, D.J. and Gilbert, S. 2005. The Development of Non-linear Numerical Model to Simulate the Behaviour of Reinforced Concrete Flat Slabs in the Vicinity of Edge Columns, *Construction and Building Materials*, 19: 703-712.

NAGFE Library. 1980. Level 0: The Science and Engineering Research Council Documentation.

Parton, G.M. and El-Barbary, M.E.S. 1983. A Finite Element Analysis for Cement Composites Sandwich Plates, *The International Journal of Cement Composites and Lightweight Concrete*, 5 (3): 181-191.

Polak, M.A. 2005. Ductility of Reinforced Concrete Flat Slab-Column Connections, *Computer-Aided Civil and Infrastructure Engineering*, 20: 184-193.

Polak, M.A. 1998. Modeling Punching Shear of Reinforced Concrete Slabs Using Layered Finite Elements, *ACI Structural Journal*, 95 (1): 71-80.

Reitman, M. and Yankelevsky, D. 1997. A New Simplified Model for Nonlinear RC Slabs Analysis, *ACI Structural Journal*, 94 (4): 399-408.

Sankarasubramanian, G. and Rajasekaran, S. 1996. Constitutive Modeling of Concrete Using a New Failure Criterion, *Computer and Structures*, 58 (5): 1003-1014.

Shanmugam, N.E., Kumar, G. and Thevendram, V. 2002. Finite Element Modelling of Double Skin Composite Slabs, *Finite Elements in Analysis and Design*, 38: 579-599.

Vainiunas, P., Popovas, V. and Vilnius, A.J. 2004. Nonlinear FEA of RC Floor Slab-to-Column Joint Connection, Theoretical Foundations of Civil Engineering-XII, Ed.by W. Szczesninak, OW PW, Warsaw, 12.

Vecchio, F. and Collins, M. 1986. The Modified Compression Field Theory for Reinforced Concrete Element Subjected to Shear, *ACI Journal*, 83 (2): 219-231.

Vecchio, F., Collins, M. and Aspiotis, J. 1994. High Strength Concrete Elements Subjected to Shear, *ACI Structural Journal*, 91 (4): 423-433.

Vidosa, F., Kotsovos, M. and Pavlovic, M. 1988. Symmetrical Punching of Reinforced Concrete Slabs: An Analytical Investigation Based on NFE Modeling, *ACI Structural Journal*, 85 (2): 241-250.

Wang, T. and Hsu, T.C.C. 2001. Nonlinear Finite Element Analysis of Concrete Structures Using New Constitutive Models, *Computer and Structures*, 79: 2781-2791.